

TEST STAND FOR TITAN 34D SRM STATIC FIRING

Vladimir Glozman
California Polytechnic University

George Shipway
Wyle Laboratories

ABSTRACT

An existing liquid engine test stand at the AF Astronautics Laboratory has been refurbished and extensively modified to accommodate the static firing of the Titan 34D solid rocket motor (SRM) in the vertical nozzle down orientation. The main load restraint structure was designed and built to secure the SRM from lifting off during the firing. In addition the structure provided weather protection, temperature conditioning of the SRM, and positioning of the measurement and recording equipment. The structure was also used for stacking/de-stacking of SRM segments and other technological processes.

The existing stand, its foundation and anchorage were thoroughly examined and reanalyzed. Necessary stand modifications were carried out to comply with the requirements of the Titan 34D SRM static firing. A new superstructure was designed and erected on top of the modified test stand. The superstructure was a steel framework consisting of corner and side columns, horizontal mounting platforms, and cross bracing elements. Mounting platform elevations and superstructure height were designed for easy modification to facilitate stacking/de-stacking procedures and potential testing of different SRMs.

The superstructure and the test stand overall were analyzed by finite element methods utilizing IMAGES-3D, ANSYS, and NASTRAN codes. Critical joints, anchors and the environmental enclosure were evaluated by conventional methods. Analyses were performed for the specified loads transmitted to the structure from the SRM, as well as for wind loads, postulated seismic disturbances, and acoustic pressure. It was determined that the test stand exhibits a sufficient factor of safety under the most conservative combination of the applied loads. Stiffness of the test structure and its dynamic properties were found to be acceptable.

Specific attention was given to the SRM mounting structure, which includes a modified thrust pylon, a newly designed pylon adapter welded into the pylon, and an aft test skirt fixture connecting the SRM to the pylon through the adapter. Detailed finite element analysis of the mounting structure with and without the SRM was performed, and response of the mounting structure to the ignition transient and other loading conditions was thoroughly investigated. It was found that stress levels in all critical elements was below the yield strength of the selected materials with sufficient safety factor. The stiffness of the mounting structure and therefore dynamic response of the SRM during the static firing was shown to be also acceptable.

The modified test stand met all of the design criteria during the successful firing of the Titan 34D SRM.

INTRODUCTION

The key element in the static firing of the Titan 34D SRM is the load restraint structure, which stabilizes the rocket and prevents it from lift-off during the test. The structure was designed and built utilizing an existing twenty year old liquid engine test stand at the AF Astronautics Laboratory. This allowed for the completion of all work for erecting a complex test facility in a timely, cost effective manner.

The existing stand was analyzed and design modifications were implemented to permit SRM testing. A new superstructure was erected on the top of the existing test stand. The superstructure consists of a steel beam framework with horizontal mounting platforms and environmental enclosure.

The ability of the existing pylon structure and its anchorage to withstand the static load prior to firing and the thrust load during firing was thoroughly investigated and necessary modifications have been implemented. A new thrust restraint fixture was designed to interface with the existing pylon structure, and to accommodate the SRM aft closure. Furthermore, the pylon and superstructure assembly were rigidly connected to provide additional stiffness for the SRM mounting. Each structure as well as the combined structure were evaluated for the specified restraint load transmitted from the SRM, as well as wind load, postulated seismic disturbances and acoustic pressure. Evaluations were performed by a combination of conventional and finite element methods using NASTRAN, ANSYS, IMAGES-3D and other structural codes. Furthermore, due to the time limitation and the critical importance of the design, evaluation of the combined structure was performed independently in two Wyle facilities (Norco, CA, and Huntsville, AL) using two different structural codes, ANSYS and NASTRAN. The results of these analyses compared favorably, and subsequent performance of the test facility during the static firing was the ultimate verification of the performed design and analytical work.

SUPERSTRUCTURE

The 6.7 m x 6.7 m x 16.5 m superstructure (Figure 1) was erected on the top of the modified existing test stand (Figure 2). The design and analysis of the superstructure was performed using the IMAGES-3D finite element program.

The main structural elements of the superstructure are:

- o Four corner columns.
- o Four side columns.
- o Three permanent horizontal mounting platforms.
- o Two removable horizontal platforms at elevations of 3.2m and 14.5m for the side restraint of 2 segment and 5 1/2 segment specimens.
- o Side diagonal cross bars.

The finite element model of the superstructure (Figure 3) is comprised of the beam structural elements identified in Table 1.

The model consists of 112 nodes which are connected with a total of 264 beam structural elements at six different cross sections, based on the anticipated design. To account for the weight of stairways, handrails, the mounting platform grating, and other nonstructural elements, the density of the structural element material was increased by 20 percent. Weather protection is provided by side panels and a roof. Wind loading on the side panels was included; stiffness of the panels was assumed to be negligible. Because the superstructure is erected on top of the existing stand, the 36 spring elements were used to represent the stiffness of the stand. The spring stiffness was determined by finite element analysis of the existing stand.

The main load conditions as the front and side wind loads of 177 km/hr (110 mph), the postulated ultimate loads on the top of the superstructure from the attached SRM, and the potential seismic load were evaluated.

The wind load was represented as a static pressure on the projected area of the four sides of the superstructure. A pressure of 250 MPa (36 ksi) applied to the facing side and a negative pressure of 192 MPa (28 ksi) on the three other sides was assumed. Two wind directions were evaluated, a front wind (road to flame side) and a side wind.

The postulated ultimate forces applied by the SRM to the top of the superstructure were comprised of two elements: two forces for a total of 144 ton in road-to-flame direction (Z) and two forces for a total of 41 ton in side-to-side direction (X).

The effect of seismic excitation was conservatively analyzed by quasistatic method applying 0.5g load in three orthogonal directions simultaneously.

An operating load of 10.6 kN applied to the mounting platform at the 10 meter elevation and self-weight of the superstructure was considered in all load cases.

Maximum deflections on the top of the superstructure were calculated to be:

- | | |
|-------------------------------|---------|
| o due to front wind load; | 6.8 mm |
| o due to side wind load; | 6.3 mm |
| o due to ultimate side forces | |
| road-to-flame direction | 15.2 mm |
| side-to-side direction | 48.3 mm |

Maximum stress due to wind load was 90 MPa which is below yield strength of A36 steel with a factor of safety of 2.8. The maximum stress caused by ultimate top forces of 180 MPa was below yield strength deflections and stresses caused by postulated seismic disturbance were substantially smaller of those due to the wind load.

In addition to the finite element analysis, conventional methods were used to address buckling of the frame elements and to evaluate joint connections. Results of these analysis conformed ability of the superstructure withstand the specified load with substantial factor of safety.

The environmental enclosure was analyzed by a combination of conventional and finite element methods and was shown to be capable of withstanding not only the specified wind load, but also the acoustic pressure during static firing of the SRM.

A dynamic analysis was performed to obtain dynamic properties of the superstructure, i.e. resonance frequencies and mode shapes, with and without the SRM. The main results of the analysis are summarized in Table 2.

EXISTING TEST STAND

The existing 1C test stand (Figure 4) was prepared by removing about 350 tons obsolete and surplus equipment including large propellant tanks, the liquid motor support and thrust measurement system, valves and piping, etc.

A preliminary analysis indicated the necessity of very limited modification of the stand main frame structure. The 12WF27 beams were changed to stronger 24WF76 beams at the superstructure interface to the stand and two other weak elements were replaced with stronger elements.

The stand was modeled as a framework (Figure 5) consisting of 321 beam elements supported at four points. The stand was analyzed for the same load conditions as the superstructure and interactive forces from the superstructure were applied to the finite element model of the stand. The minimum factor of safety for the wind load was calculated to be more than six against the yield strength of A36 steel.

The maximum stresses in the stand caused by the abnormal ultimate load applied to the top of the superstructure was calculated to be 345 MPa which was below ultimate strength of A36 steel.

The modified stand was also analyzed for a vertical downward load to determine its vertical load carrying capacity at the superstructure boundary. A factor of safety of 4.6 against ultimate strength was calculated for that load condition.

Stiffness properties of the existing stand also were calculated and subsequently used for a pylon-to-stand connection evaluation and for creating a general combined finite element model.

MOTOR MOUNTING STRUCTURE

The attachment structure for the mounting of the motor (Figure 6) includes an existing pylon, newly designed pylon adapter welded into the pylon, and an aft test-skirt fixture connecting the motor to the pylon through the pylon adapter. A finite element analysis of the pylon structure, including a new pylon adapter, skirt, and a simple (stick) model of the SRM was performed using the IMAGES-3D and NASTRAN finite element programs. The model of the pylon is comprised entirely of beam elements. This assumes that cross sections, particularly those of the box beam at the top, do not warp. All beam connections are assumed to be rigid, as are the four support points. All section properties are based on the drawings of the original pylon structure.

The pylon adapter model is composed primarily of quadrilateral and triangular plate elements. The skirt model is basically a cylinder with a stiffening ring at the bottom. The SRM is modeled as a beam with section properties based on actual specimen geometry. The model was subjected to gravity loads in the three orthogonal directions. This served as a check of the structural soundness of the model, and also provided information for evaluating the structures' response to potential earthquake excitation.

The dynamic analysis of the mounting structure with and without the SRM was performed first in order to obtain the response of the structure to the ignition transient. The lowest resonance frequency of the SRM itself as a simple cantilever beam was computed at 1.96 Hz. The lowest resonance frequencies of the SRM installed are 1.22 Hz and 1.30 Hz in side-to-side and road-to-flame direction correspondingly. The dynamic response of the loaded structure to the ignition transient showed that there was no amplification of the vertical component of the motor displacement. However, the horizontal component was 1.7 times the static. The dynamic load factor was used to adjust the static force for a load combination calculation. Detailed static stress analysis showed that the only significant stress of 140 MPa occurs in the pylon adapter upper ring. Stresses in other parts of the pylon and the adapter were substantially lower. The maximum stress in the aft test skirt caused by the abnormal "ultimate" thrust of 10 million pounds canted at six degrees, was calculated to be 490 MPa. This is below the tensile strength of 620 MPa for normalized 4130 alloy steel.

PYLON ANCHOR YIELD ANALYSIS

Pylon anchor yield analysis was critical. It preceded all other analyses, because the existing 40 steel anchors were imbedded in concrete and, therefore, could not be replaced. Each anchor is 57.1 mm (2.25 in.) in diameter and 11.6 m long and is made of quenched and tempered 4340 steel. Detailed analysis was performed taking into account anchor/concrete relative stiffness and the anchor/concrete interface. Even though the anchors can withstand the abnormal "ultimate" thrust of 10 million, maximum thrust force must be limited to 8.6 M based on yield of the anchors, and to only 7.2 M to maintain the joint. But even the smallest allowable thrust is substantially higher than that predicted and recorded during the SRM test.

COMBINED MODEL ANALYSIS OF THE TEST STAND

After design and modification of the main elements of the test stand were completed, the finite element analyses of the stand were carried out. A combined finite element model of the superstructure, test stand, pylon assembly and SRM were developed on the NASTRAN and ANSYS finite element computer codes and carried out by two independent teams. These analyses were aimed to verify the design effort and also examine the stand-pylon-SRM interfaces. Therefore, the three following cases were analyzed:

1. Free standing pylon with the SRM without any attachment to the restraining structure.
2. Pylon with the SRM attached to the superstructure at the top level by means of struts.

3. Pylon attached to the stand at the second platform level with the SRM attached to the superstructure at the top level.

The analyses of the combined model were performed for different load conditions: thrust load, wind load, seismic disturbances, etc. The analyses took into account the transient nature of the thrust force and the SRM mass decay during the burnout.

The results of the combined model analyses gave additional information about static and dynamic behavior of the stand. They were in good agreement with the previously obtained data, and therefore, verified overall new design and modification of the test stand and the pylon.

Performance of the test stand during the SRM static firing was in good correlation with analytical predictions. No trace of damage, malfunction, or any structural problems have been noticed during the test. The measured stresses, eigenvalues, and eigenvectors were very close to those obtained during design and analytical activities. Overall the test stand performed exceptionally well and met all design criteria during the successful firing of the Titan 34D SRM.

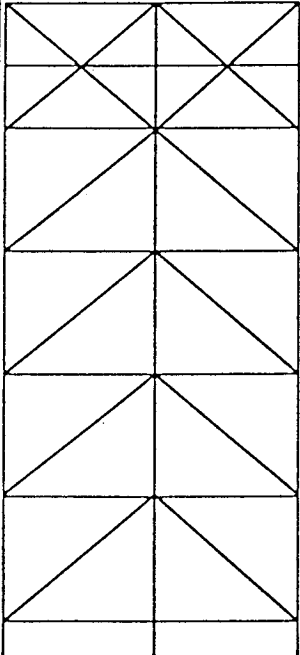
LIST OF FIGURES

1. Superstructure framework.
2. Main structure of the test stand for Titan 34D SRM firing.
3. Finite element model of the superstructure.
4. Test stand 1C.
5. Finite element model of the modified test stand 1C.
6. Finite element model of motor mounting structure.

LIST OF TABLES

1. Superstructure node and beam element summary.
2. Resonance frequencies of superstructure, Hz.

TABLE 1
SUPERSTRUCTURE NODE AND BEAM ELEMENT SUMMARY

ELEV. M	VIEW	NODE NUMBER	COLUMN		PLATFORM W12 x 26				SIDE
			CORNER W12 X 58	SIDE W12 X 26	INNER	OUTER	CORNER	CROSS	Two 6X4X1/2
16.2		92-104	25-28	53-56	—	185-192	—	—	
14.5		79-91, 105-112	21-24	19-52	157-164	165-180	153-156	197-200	233-264
12.8		66-78	17-20	45-48	—	145-152	—	—	225-232
10.0		53-65	13-16	41-44	117-124	125-132	133-136	—	217-244
6.7		40-52	9-12	37-40	97-104	105-112	113-116	—	209-216
3.2		27-39	5-8	33-36	77-84	85-92	93-96	192-196	201-208
0.1		14-26 1-13	1-4	29-32	57-64*	65-72*	—	—	
0									

* SECTIONS W16 X 67

TABLE 2
 RESONANCE FREQUENCIES OF SUPERSTRUCTURE, HZ

Mode Description	Without Specimen	With Specimen
Global rocking side-to-side	6.67	2.58
Global rocking road-to-flame	6.45	2.59
Global torsion		7.41
Local side column bending	4.29	4.29
Lower platform (local)	8.66, 8.72	8.66, 8.71
Upper platform (local)	8.72, 8.73	8.71, 8.73, 8.75

ORIGINAL PAGE IS
OF POOR QUALITY

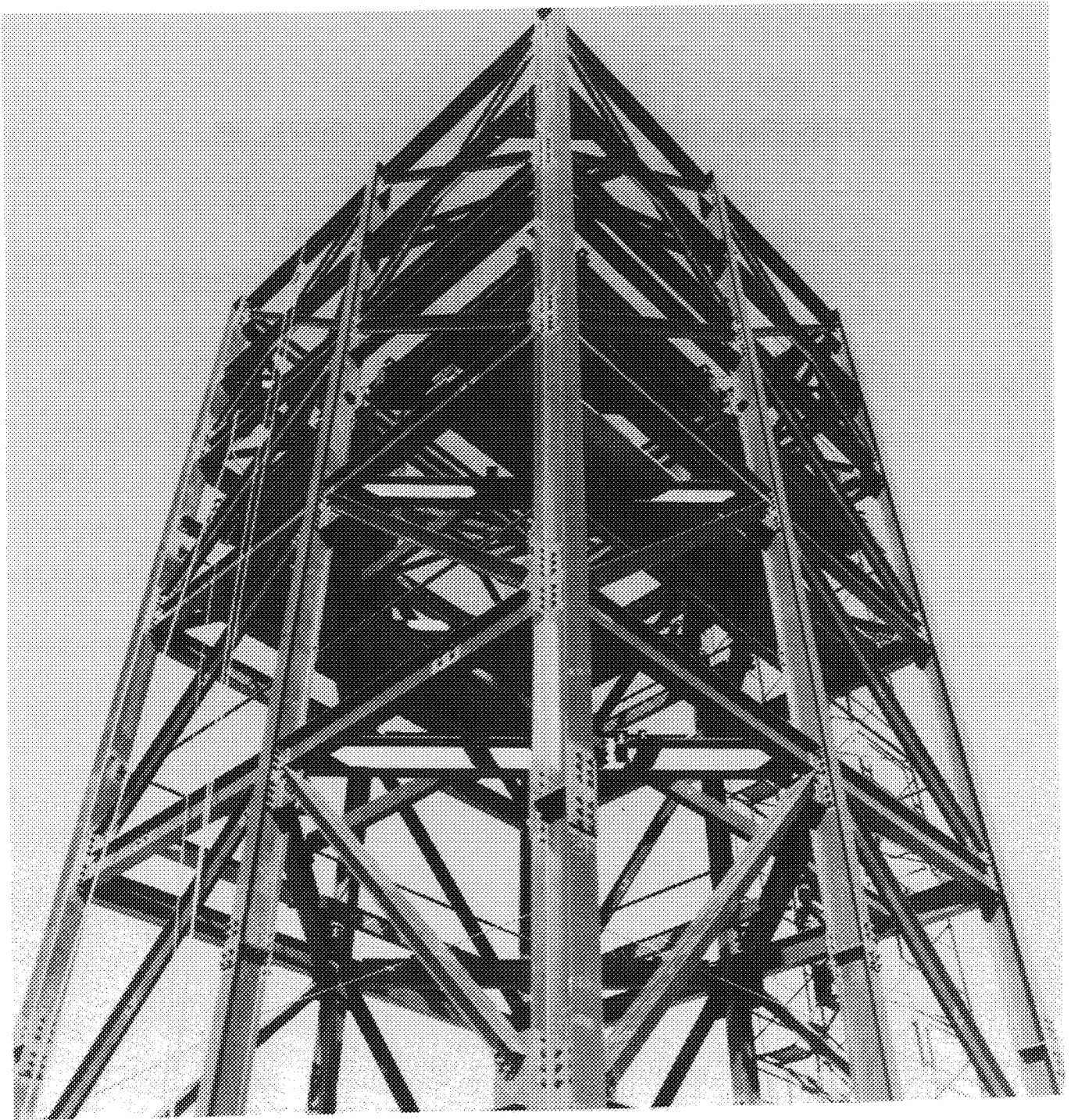


FIGURE 1
SUPERSTRUCTURE FRAMEWORK

ORIGINAL PAGE IS
OF POOR QUALITY

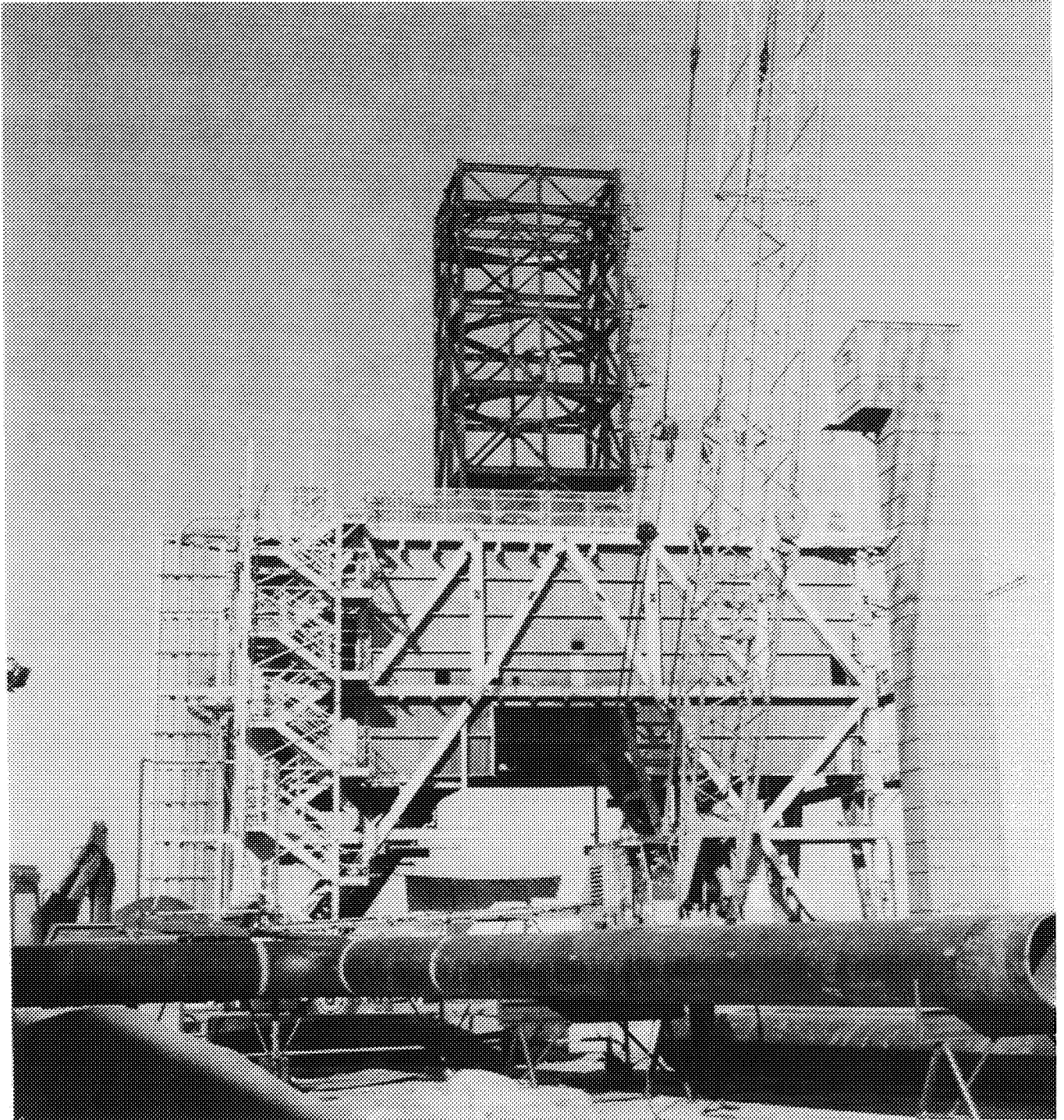


FIGURE 2

MAIN STRUCTURE OF THE TEST STAND FOR TITAN 34D SRM FIRING

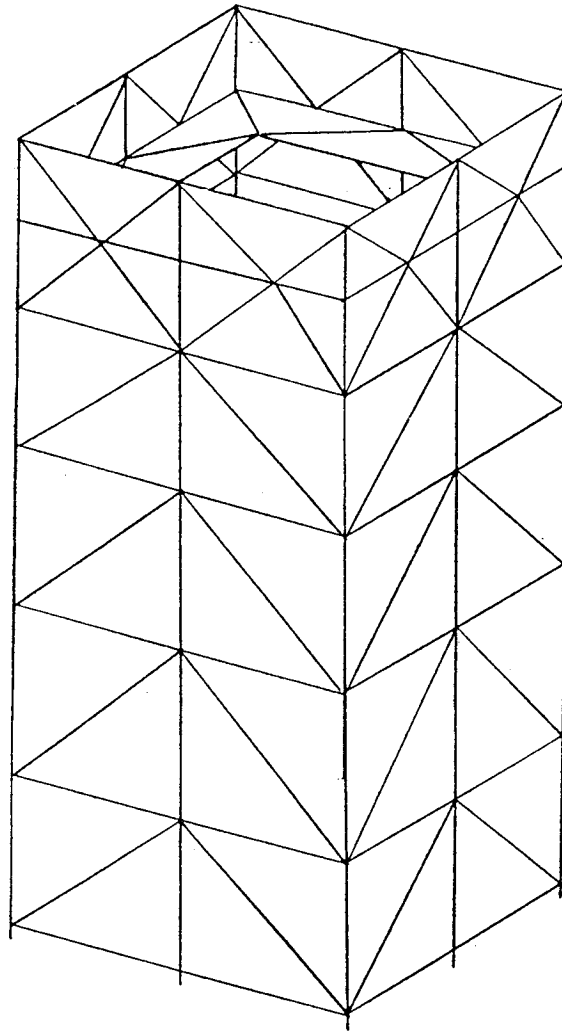


FIGURE 3
FINITE ELEMENT MODEL OF THE SUPERSTRUCTURE

ORIGINAL PAGE IS
OF POOR QUALITY

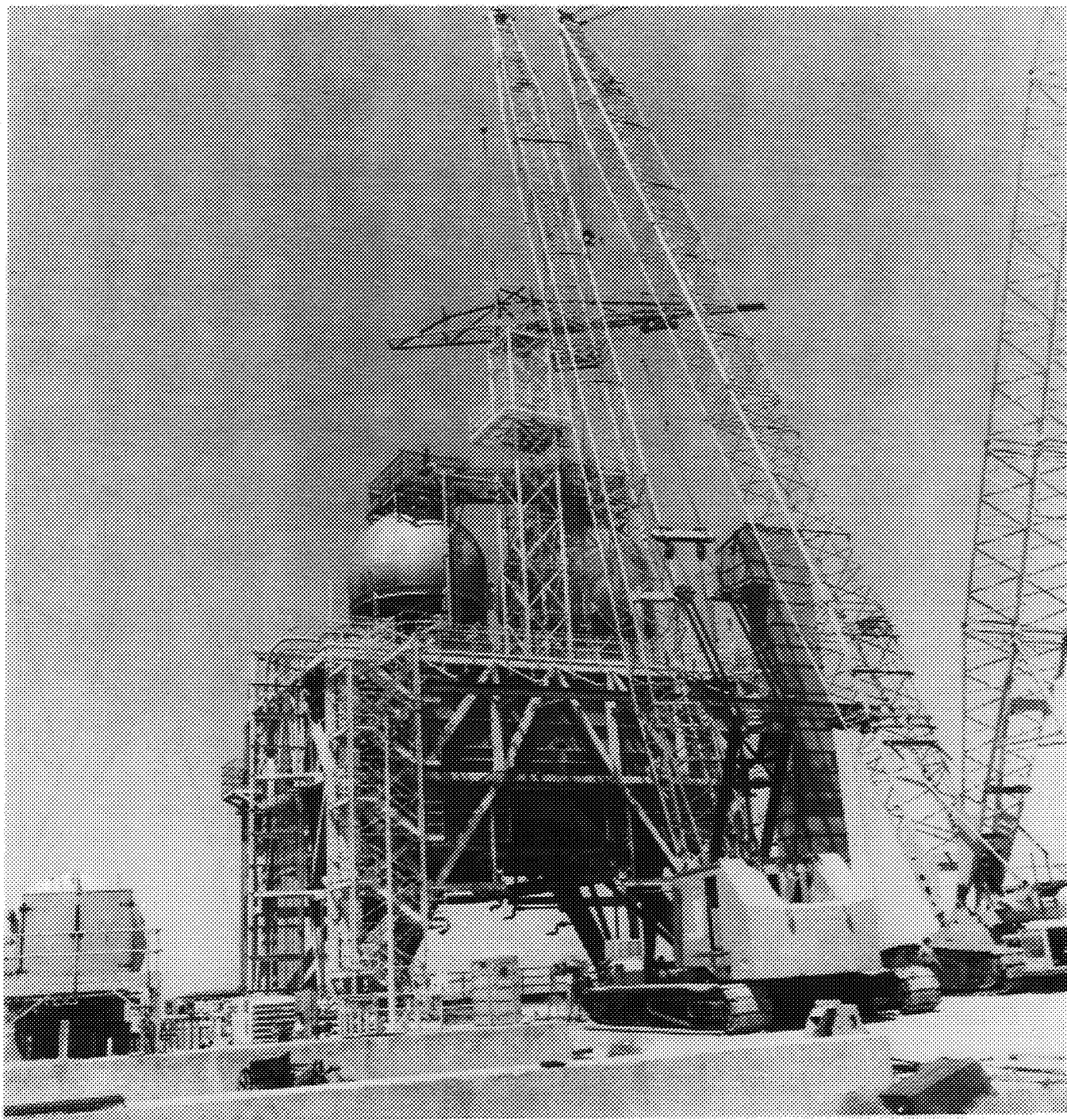


FIGURE 4
TEST STAND 1C

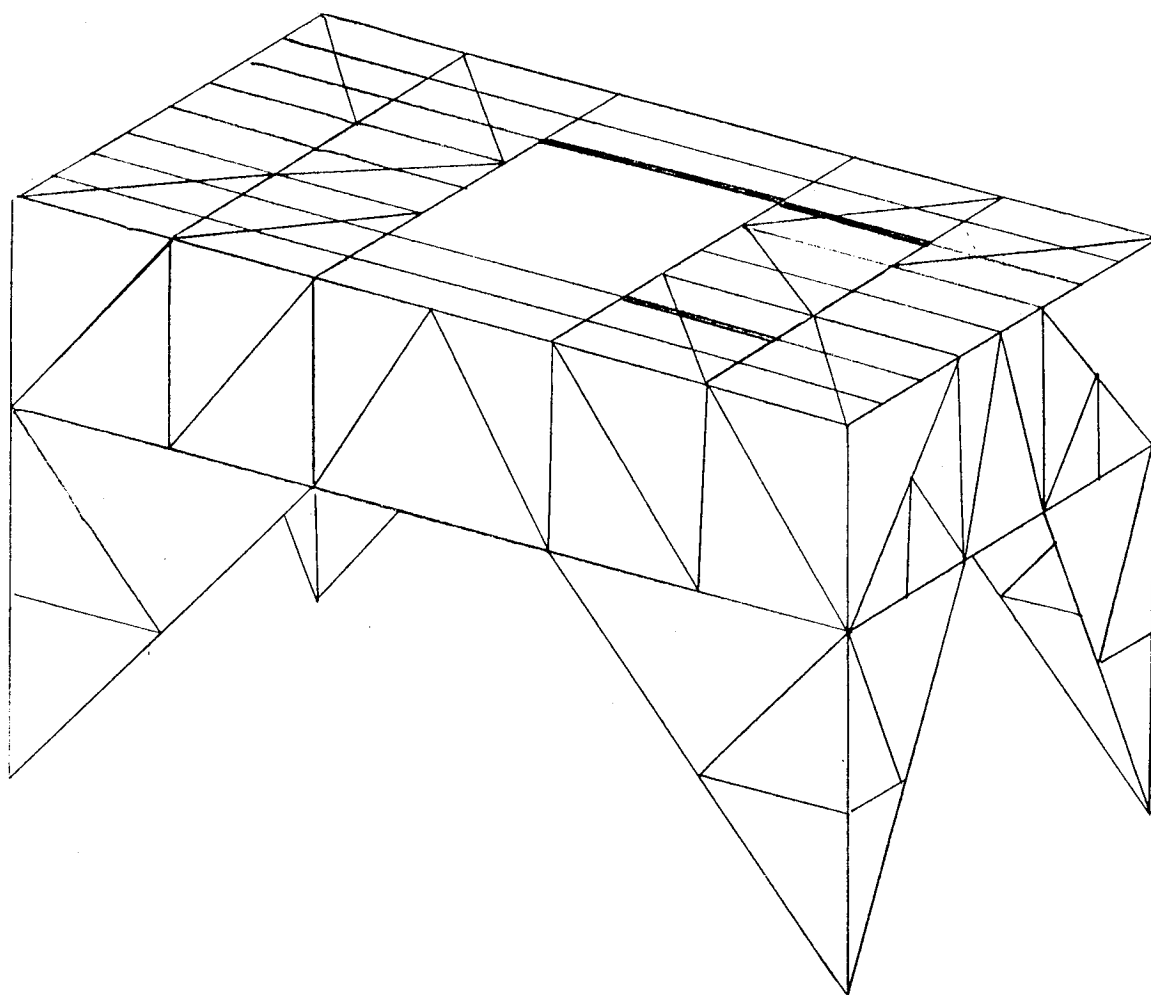


FIGURE 5

FINITE ELEMENT MODEL OF THE MODIFIED TEST STAND 1C

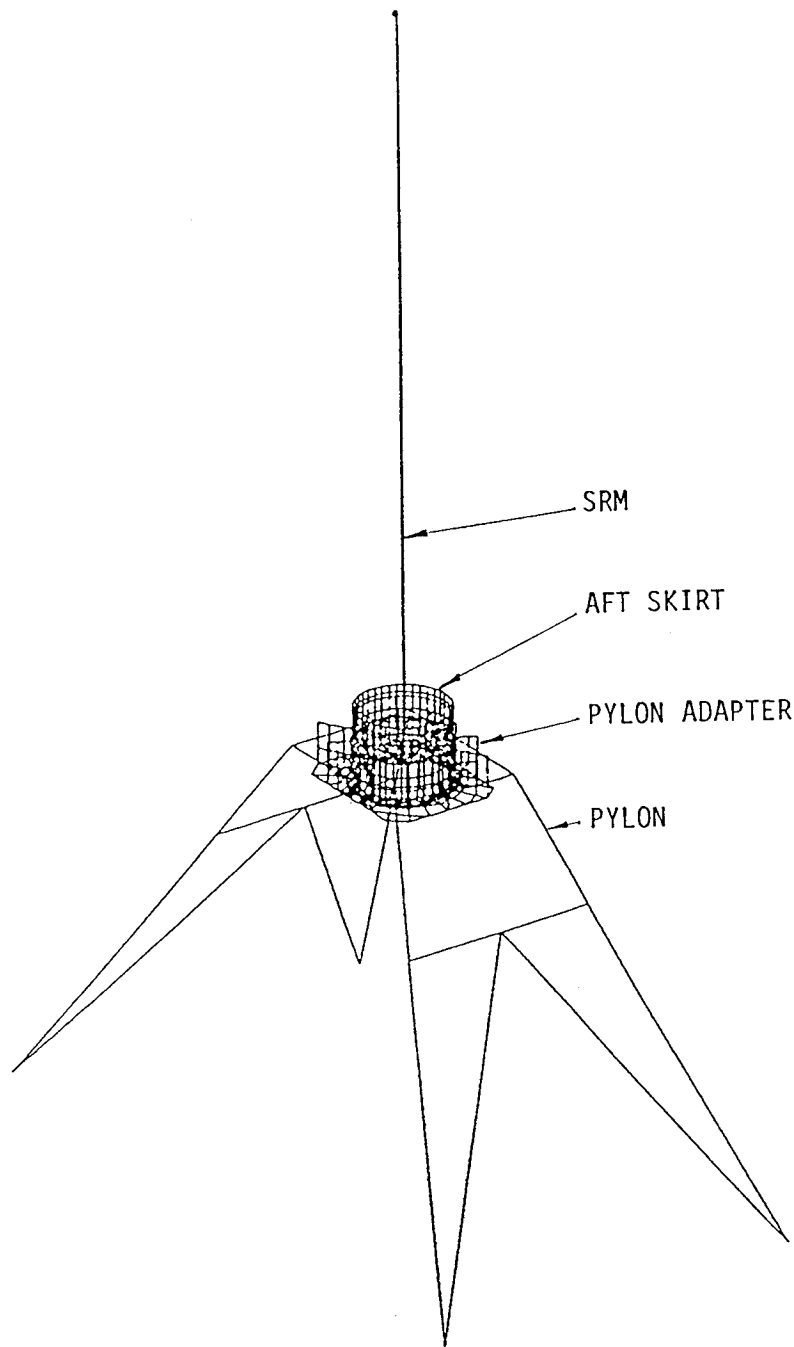


FIGURE 6

FINITE ELEMENT MODEL OF MOTOR MOUNTING STRUCTURE